



# CFD Simulation on the J-2X Engine Exhaust in the Center-Body Diffuser and Spray Chamber at the B-2 Facility

*Xiao-Yen Wang, Thomas Wey, and Robert Buehrle  
Glenn Research Center, Cleveland, Ohio*

## NASA STI Program . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI Program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Reports Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.
- **CONFERENCE PUBLICATION.** Collected

papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services also include creating custom thesauri, building customized databases, organizing and publishing research results.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to [help@sti.nasa.gov](mailto:help@sti.nasa.gov)
- Fax your question to the NASA STI Help Desk at 301-621-0134
- Telephone the NASA STI Help Desk at 301-621-0390
- Write to:  
NASA Center for AeroSpace Information (CASI)  
7115 Standard Drive  
Hanover, MD 21076-1320



# CFD Simulation on the J-2X Engine Exhaust in the Center-Body Diffuser and Spray Chamber at the B-2 Facility

*Xiao-Yen Wang, Thomas Wey, and Robert Buehrle  
Glenn Research Center, Cleveland, Ohio*

Prepared for the  
2008 Thermal and Fluids Analysis Workshop  
cosponsored by the NASA AMES Research Center and San Jose State University  
San Jose, California, August 18-22, 2008

National Aeronautics and  
Space Administration

Glenn Research Center  
Cleveland, Ohio 44135

## Acknowledgments

The first author would like to thank Kevin Dickens and Daryl Edwards for their valuable input to this work.

This report contains preliminary findings,  
subject to revision as analysis proceeds.

*Level of Review:* This material has been technically reviewed by technical management.

Available from

NASA Center for Aerospace Information  
7115 Standard Drive  
Hanover, MD 21076-1320

National Technical Information Service  
5285 Port Royal Road  
Springfield, VA 22161

Available electronically at <http://gltrs.grc.nasa.gov>

# **CFD Simulation on the J-2X Engine Exhaust in the Center-Body Diffuser and Spray Chamber at the B-2 Facility**

Xiao-Yen Wang, Thomas Wey, and Robert Buehrle  
National Aeronautics and Space Administration  
Glenn Research Center  
Cleveland, Ohio 44135

## **Abstract**

A computational fluid dynamic (CFD) code is used to simulate the J-2X engine exhaust in the center-body diffuser and spray chamber at the Spacecraft Propulsion Facility (B-2). The CFD code is named as the space-time conservation element and solution element (CESE) Euler solver (ref. 1) and is very robust at shock capturing. The CESE results are compared with independent analysis results obtained by using the National Combustion Code (NCC) (ref. 2) and show excellent agreement.

## **Introduction**

The B-2 in the Plum Brook Station (PBS) was originally designed to test full-scale upper-stage rockets up to 100,000 lbf thrust in a simulated space environment. Since most rocket engines that have been tested in the B-2 were in the 30,000 lbf thrust class, the B-2 must be adapted to accommodate the engines with much larger thrust such as the J-2X engine that produces 294,000 lbf thrust, which results in a more severe thermal environment and a larger scale of energy.

A sketch of B-2 facility is shown in figure 1. The J-2X engine exhaust was directed into the center-body diffuser to slow down before hitting the top surface of the water tank at the bottom of the spray chamber. When the rocket engine is operating, the water is injected inside the spray chamber to cool down the hot exhaust gas. The mixture of water vapor and hot gas will vent through the ejector when the spray chamber pressure is high enough. The steam is sprayed through the steam blocker to prevent the back flow in the event of J-2X engine shutdown. A CFD code is used to simulate how the J-2X engine exhaust expands through the center-body diffuser and into the spray chamber, then vents to outside the chamber through the ejector. The water spray inside the spray chamber is not modeled here. The two-dimensional/axisymmetric CESE Euler code is used here. In the following, the J-2X engine performance is described first, which is followed by the CFD results and validations.

## **J-2X Engine Performance**

The J-2X engine uses liquid oxygen ( $O_2$ ) and hydrogen ( $H_2$ ) as a propellant with an oxidizer to fuel (o/f) ratio of 5.5 at the chamber pressure of 1,338 psia to produce 294,000 lbf of thrust. The chemical equilibrium compositions and applications (CEA) code is used to compute the performance of the J-2X engine. In table 1, the CEA results of the pressure ( $p$ ), temperature ( $T$ ), density ( $\rho$ ), mole weight, ratio of specific heat ( $\gamma$ ), sonic velocity, Mach number, and mole fractions are listed for different locations inside the J-2X engine.

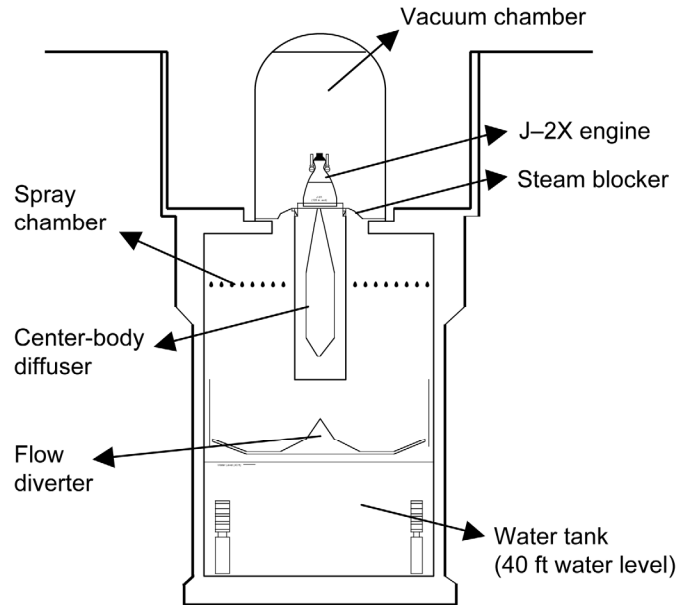


Figure 1.—A sketch of the B-2 facility for J-2X engine testing.

TABLE 1.—J-2X ENGINE PERFORMANCE (CEA RESULTS)

	Combustor end	Throat	Exit
$p$ , BAR	85.62	51.066	0.0592
$T$ , K	3406.75	3210.22	1005.01
$\rho$ , kg/m <sup>3</sup>	3.8389	2.4501	9.28E-03
Mole weight, (1/n)	12.7	12.806	13.103
$C_p$ , kJ/kg-K	7.52	6.81	2.895
$\gamma$	1.15	1.15	1.28
Sonic velocity, m/s	1601.3	1549.8	903.7
Mach number	0.26	1	4.92
Mole fractions			
H <sub>2</sub>	0.301	0.301	0.307
H <sub>2</sub> O	0.64	0.655	0.693

## CFD Simulation on the J-2X Engine Exhaust

It was assumed that the hot gas is an ideal gas. The hot gas properties at the engine nozzle exit obtained from CEA is used in the CFD simulation. The CESE two-dimensional/axisymmetric Euler code is used and a finite-element mesh is generated using MSC Patran. In the analysis, the flow variables are nondimensionalized by using those at the engine nozzle exit as follows:

$$\begin{aligned}
 p^* &= p / \rho_e u_e^2, & \rho^* &= \rho / \rho_e \\
 x^* &= x / L, & T^* &= T / T_e \\
 u^* &= u / u_e, & t^* &= t / (L / u_e)
 \end{aligned}$$

where  $\rho_e = 9.94e-3 \text{ kg/m}^3$ ,  $u_e = 4,446.2 \text{ m/s}$ , and  $T_e = 1,005 \text{ K}$  are the density, velocity, and temperature, and

$$L = 1 \text{ m and } \rho_e u_e^2 = 1.965 \times 10^5 \text{ N/m}^2 = 28.5 \text{ psia.}$$

The computational domain that has 18,119 mesh points and 35,332 triangular elements is shown in figure 2. The water surface is approximated by using a solid wall. For the core flow, it was assumed that the total pressure  $p_t = 1,338$  psia and the total temperature  $T_t = 3,552$  K (5,934 °F) with a mass flow rate of 650 lbm/s. For the steam blocker,  $p_t = 165$  psia,  $T_t = 459$  K (366 °F) with a mass flow rate of 147 lbm/s. The initial conditions at  $t = 0$  (B-2 evacuated conditions) are defined as

$$p^* = 0.0056 \text{ (} p = 0.16 \text{ psia)}, u^* = 0, v^* = 0, \rho^* = 0.8047$$

At the inlet of the computational domain (exit of the engine nozzle)

$$p^* = 0.0368, u^* = 0.9957, v^* = 0, \rho^* = 1.1092$$

At the inlet of steam blocker

$$p^* = 0.00723, u^* = 0.7028, v^* = 0, \rho^* = 0.7313$$

At the opening to the ejector

$$p_{\text{back}}^* = 0.014 \text{ (} p_{\text{back}} = 0.4 \text{ psia)}$$

The computed CESE results of the nondimensional density, pressure, temperature, Mach number, and velocity vector at  $t = 0.0787$  s are plotted in figure 3. It can be seen that complex shock waves exist inside both the diffuser and the spray chamber. In the center-body diffuser, a series of oblique shock waves start at the exit of the engine nozzle (inlet of the computational domain) and keep reflecting along the wall. The flow field inside the diffuser reaches steady state within 0.0787 s. The flow at the exit of the diffuser is still supersonic. The shock waves in the spray chamber still bounce back and forth along the chamber wall and water surface.

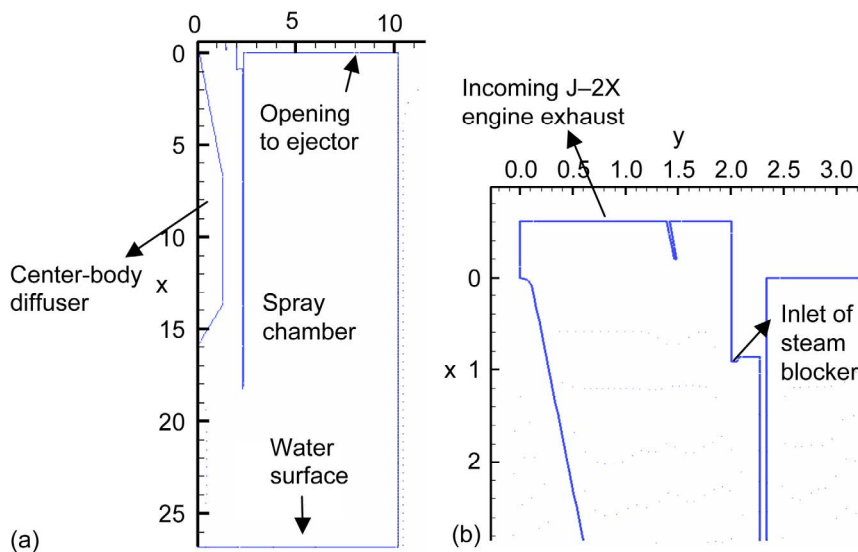


Figure 2.—Computational domain used in the analysis. (a) The computational domain. (b) Close view of the domain inlet.

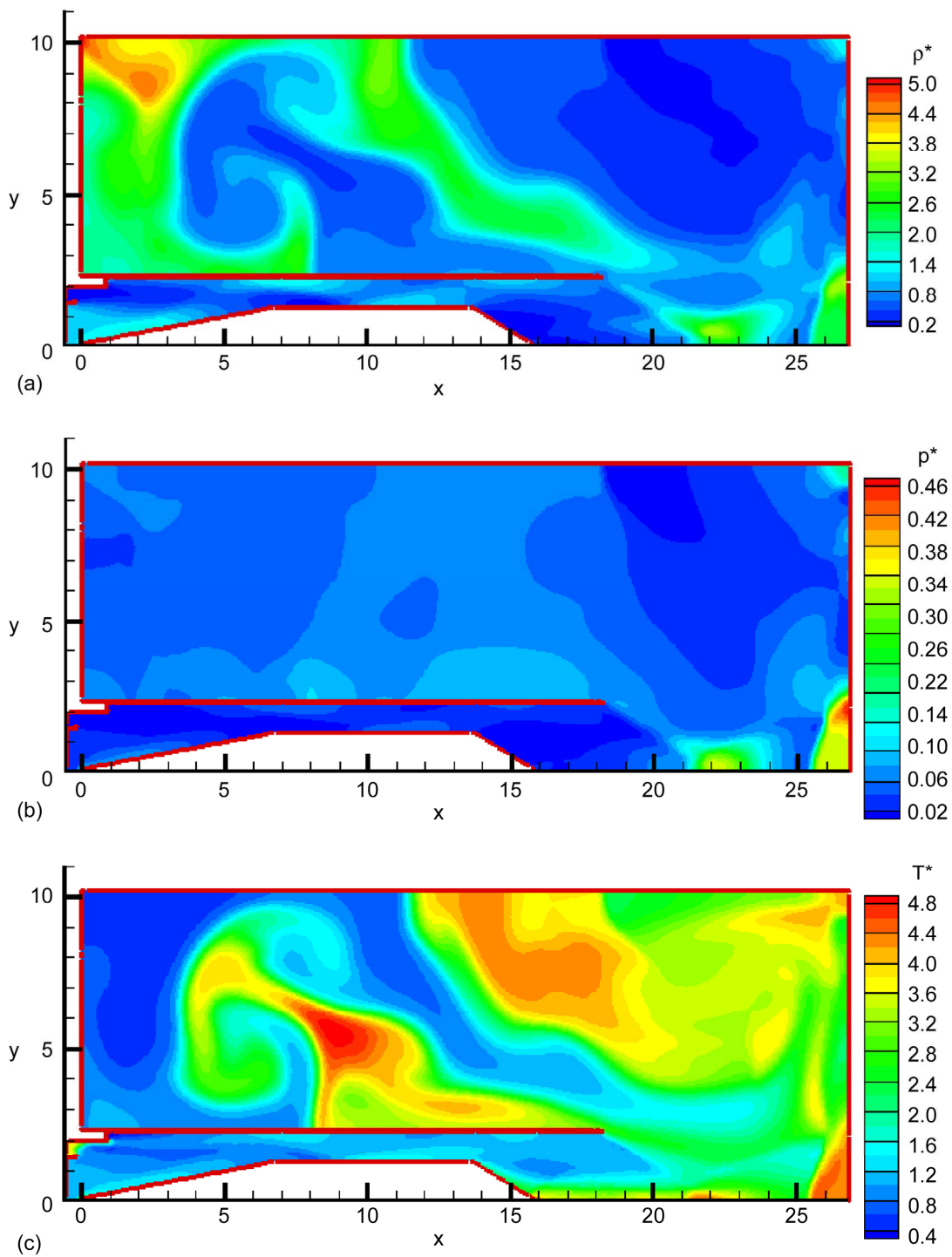


Figure 3.—CESE results at  $t = 0.0787$  s.



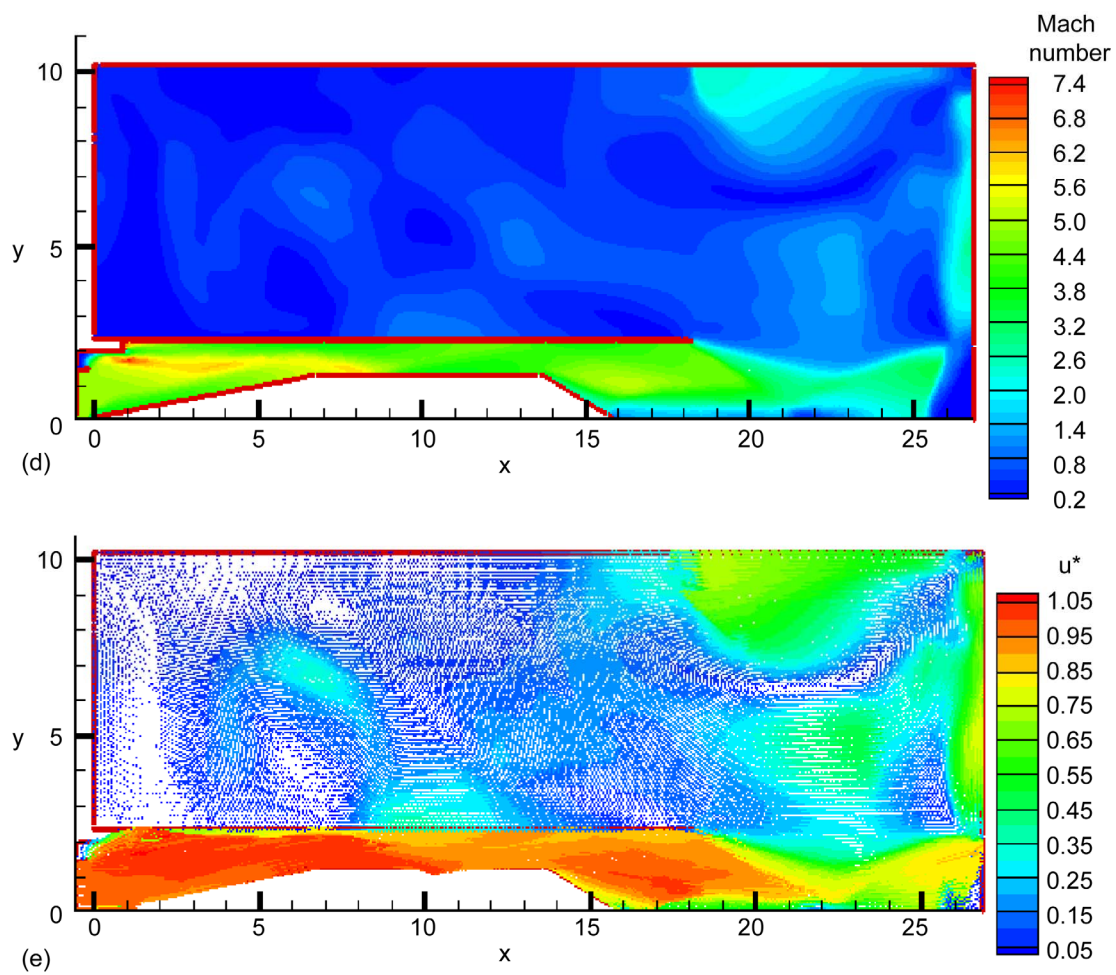


Figure 3.—CESE results at  $t = 0.0787$  s, concluded.

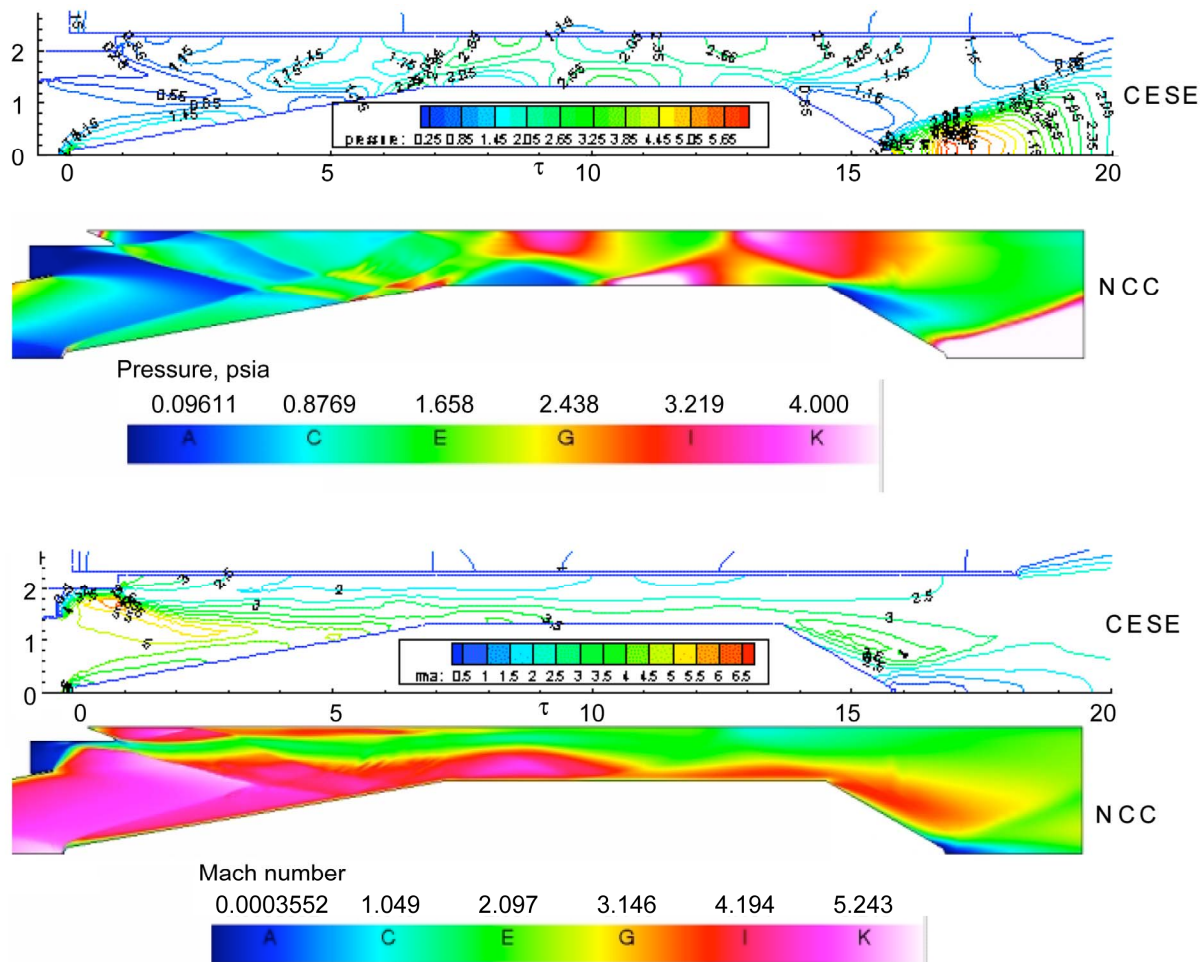


Figure 4.—The comparison between CESE and NCC results inside the center-body diffuser.

Further, the flow field inside the center-body diffuser is compared between CESE results and those computed independently using the NCC. The details of the NCC simulation will be given in a separate paper and will not be described here. It can be seen that the wave patterns captured in the NCC and CESE codes are very similar as shown in figure 4.

## Conclusions

The J-2X engine exhaust in the center-body diffuser and spray chamber at the Spacecraft Propulsion Facility (B-2) is simulated using the CESE method. The shock wave pattern was captured by the CESE method and agrees well with the corresponding results obtained by using the NCC. Further analysis is needed to validate the design of the B-2 for testing rocket engines with up to 300,000 lbf thrust.

## References

1. X.Y. Wang, "Space-time CESE Euler Solver 6.0, User's Guide," July, 2001.
2. R.M. Stubbs and N.S. Liu, "Preview of National Combustion Code," AIAA-97-3114, 1997.

REPORT DOCUMENTATION PAGE				Form Approved OMB No. 0704-0188	
<p>The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.</p> <p>PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.</p>					
1. REPORT DATE (DD-MM-YYYY) 01-01-2009		2. REPORT TYPE Technical Memorandum		3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE CFD Simulation on the J-2X Engine Exhaust in the Center-Body Diffuser and Spray Chamber at the B-2 Facility				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) Wang, Xiao-Yen; Wey, Thomas; Buehrle, Robert				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER WBS 931164.03.01.02.01	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191				8. PERFORMING ORGANIZATION REPORT NUMBER E-16610	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001				10. SPONSORING/MONITORS ACRONYM(S) NASA	
				11. SPONSORING/MONITORING REPORT NUMBER NASA/TM-2009-215464	
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Categories: 20 and 34 Available electronically at <a href="http://gltrs.grc.nasa.gov">http://gltrs.grc.nasa.gov</a> This publication is available from the NASA Center for AeroSpace Information, 301-621-0390					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT A computational fluid dynamic (CFD) code is used to simulate the J-2X engine exhaust in the center-body diffuser and spray chamber at the Spacecraft Propulsion Facility (B-2). The CFD code is named as the space-time conservation element and solution element (CESE) Euler solver and is very robust at shock capturing. The CESE results are compared with independent analysis results obtained by using the National Combustion Code (NCC) and show excellent agreement.					
15. SUBJECT TERMS CFD simulation; J-2X engine exhaust					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES 12	19a. NAME OF RESPONSIBLE PERSON
a. REPORT U	b. ABSTRACT U	c. THIS PAGE U			STI Help Desk (email: <a href="mailto:help@sti.nasa.gov">help@sti.nasa.gov</a> )
					19b. TELEPHONE NUMBER (include area code) 301-621-0390



